Renewable and Sustainable Energy Reviews 25 (2013) 122–134

Contents lists available at SciVerse ScienceDirect

Renewable and Sustainable Energy Reviews

journal homepage: www.elsevier.com/locate/rser

Review of computer-aided numerical simulation in wind energy

Aaron Miller [a], Byungik Chang [a,*], Roy Issa [b], Gerald Chen [b]

[a] *Alternative Energy Institute, West Texas A&M University, Canyon, TX, USA*
[b] *School of Engineering and Computer Science, West Texas A&M University, Canyon, TX, USA*

ARTICLE INFO

Article history:
Received 21 August 2012
Received in revised form
14 March 2013
Accepted 18 March 2013
Available online 20 May 2013

Keywords:
CFD
FEA
Numerical simulation
Wind energy
Wind turbine
Computer modeling
Review

ABSTRACT

Many advances have been made during the last decade in the development and application of computational fluid dynamics (CFD), finite element analysis (FEA), numerical weather modeling, and other numerical methods as applied to the wind energy industry. The current information about this area of study may help researchers gage research efforts. Specifically, micro-siting, wind modeling and prediction, blade optimization and modeling, high resolution turbine flow modeling, support structure analysis, and noise prediction have been the main focuses of recent research. The advances in this area of research are enabling better designs and greater efficiencies than were possible previously. The trends toward system coupling, parallel computing, and replacing experiments are discussed. The shortcomings of recent research and areas of possible future research are also presented.

Published by Elsevier Ltd.

Contents

1. Introduction

Numerical simulation has become a valuable asset to research in wind energy and helped the industry become more efficient and productive. It has enabled new designs and levels of efficiency not

* Correspondence to: School of Engineering and Computer Science, Alternative Energy Institute, West Texas A&M University, Canyon, TX, USA. Tel.: +1 806 651 2295; fax: +1 806 651 2733.
E-mail address: bchang@wtamu.edu (B. Chang).

1364-0321/$ - see front matter Published by Elsevier Ltd.
http://dx.doi.org/10.1016/j.rser.2013.03.059

possible before [1,2]. Much research is being done to improve the accuracy and applicability of numerical tools. Researchers need to understand the current status and trends within this area of study to gage research projects. The recent articles in this paper came from relevant journals, and are presented to explore advances and trends in numerical simulation as applied to wind energy to satisfy this need. Micro-siting, wind modeling and prediction, blade optimization and modeling, high resolution turbine flow modeling, support structure analysis, and noise prediction are the main focuses of the studies presented here.

Determining the exact placement of wind turbines in a wind farm, or micro-siting, used to consist of guess work and simple models, but numerical methods are enabling optimization of placement and better understanding of wake interactions of turbines. Without these recently possible techniques, wind energy could not be harnessed to its full potential. The atmospheric boundary layer (ABL) also was the subject of some speculation, but measurements coupled with new numerical techniques have made accurate determination of a site's wind energy potential possible. Much advancement has been made in these areas, but there is plenty of room for improvement. Improvements could be made not only through new techniques and methods, but also through combinations and optimizations of recently devised models.

Advancements have been made in large-scale applications, such as the aforementioned, but smaller-scale and more detailed models also have been the subject of recent research. CFD can be employed alone or coupled or in conjunction with existing structural analysis tools to analyze, optimize, or create blade designs for wind turbines. Force and fatigue analysis, power output optimization, and damage prediction for complex, modern blades are now possible because of the recently developed numerical techniques. Consideration of detailed models of blades, generators, and support structures simultaneously is now possible. Prediction of noise generation and minimizing noise are even possible.

All of the methods outlined in this paper have been made possible by increases in computing power and improvements in computing techniques. A technical improvement is coupling CFD, FEA, and other numerical tools together to utilize the strength of each. Increasing computing power is not limited to a single, local system. Multiple, parallel, dedicated server arrays have been created and utilized for demanding, large CFD solutions, enabling faster, larger, and more detailed results than ever before.

2. Turbine micro-siting and wind farm layout optimization

Wind farm micro-siting is the process of determining the most efficient and economical configuration for wind turbines within a wind farm. Researchers recently have applied numerical simulation techniques to this process. These new techniques have helped optimize and analyze situations not possible before, such as flow in complex terrain [3] and cost optimization models [2].

2.1. Wind flow analysis

Computational fluid dynamics (CFD) and numerical methods can be used to analyze flow over complex terrain in order to better micro-site a wind farm. Coastal areas are attractive farm locations because of the relatively high wind speeds, but these and other attractive regions often have complex terrain and unsteady wind flow. CFD and numerical modeling aid, and sometimes make possible, micro-siting in these areas. As an example, Palma et al. [4] analyzed flow in such a region with CFD, and results were accurate when compared to observations and experiments. Their method predicted regions where flow was unsteady, separated, or had too much vorticity for turbines. Their CFD setup and meshing are shown in Fig. 1. A similar study used custom code based on large eddy simulation (LES) to analyze flow over complex terrain [3]. The custom method was accurate in predicting wake regions behind hills and other features.

Micro-siting in an urban environment with buildings is another focus of recent research. Ledo et al. [5] employed the Reynolds-Averaged Navier–Stokes (RANS) method to perform such an analysis. They found flat-roofed buildings best for turbines. The model used a semi-log wind profile to simulate realistic wind. While analysis of complex terrain is useful in micro-siting, considering the effects of the turbines on the atmospheric boundary layer (ABL) and each other is important.

Turbines and the ABL interact in the real world, and the effects of this interaction can significantly impact farm performance. Traditional wall roughness models are used to estimate the effect of turbines on the ABL, but Johnstone and Coleman [6] used a large array of turbines to simulate this effect more accurately. They used the actuator disc method to approximate turbine geometry. The study found that turbines of mixed height may be more efficient.

These recent studies have provided useful methods for finding suitable locations for wind turbines in complex environments, but most of those considered here do not include turbine geometry in their models. Including turbine geometry and using more advanced methods, such as LES, could be the focus of further research. Table 1 compares the presented research in wind flow analysis by listing the employed methods and programs, major contribution of the study to the field, the blade model, and results. Having feasible and accurate methods to simulate the wind and blade geometry improve farm efficiency, but a good engineering design model considers many more variables.

2.2. Algorithmic and cost-analyzing methods

CFD can be coupled with algorithms which use cost, spacing, geography, and turbine interconnections as variables in order to find the optimum farm layout. Models considering a more complete list of variables can optimize and predict, not simply analyze, wind farm layouts. One such model approaches micro-siting as a constrained optimization problem where power must be maximized and distance between turbines minimized [7]. The model employs a Gaussian particle swarm optimization algorithm. Only one type of turbine at one height can be analyzed, and cost and other restrictions are not considered. Torres et al. [8] used a similar model in the CFD program EllipSys3D, employing LES and the SIMPLE algorithm, to optimize the layout of a wind farm. They used the actuator disc model and analyzed wake–wake interactions. Another optimization study [9] treated wakes as particles generated by turbines. The particles flow in a pre-calculated flow field, and the decrease in velocity due to wake interaction is analyzed. Since the flow field is pre-calculated, no new CFD calculations need to be done for each new layout. Fig. 2 shows an example of an optimized wind farm layout on a topographical map produced by this method. These models can optimize power output, but may not necessarily produce the most economical layout. This goal requires a cost model.

Numerical models which use cost as a variable increase a farm's return and help make wind energy more competitive. One optimization study defined cost in terms of rotor diameter [10]. The model simultaneously selects the optimal rotor diameter and turbine locations for maximum power generation. The model can improve power output significantly, but does not consider hub height. A more advanced model [2] uses CFD wake analysis in conjunction with an evolutionary algorithm for micro-siting based on geography, wind data, installation cost, and turbine

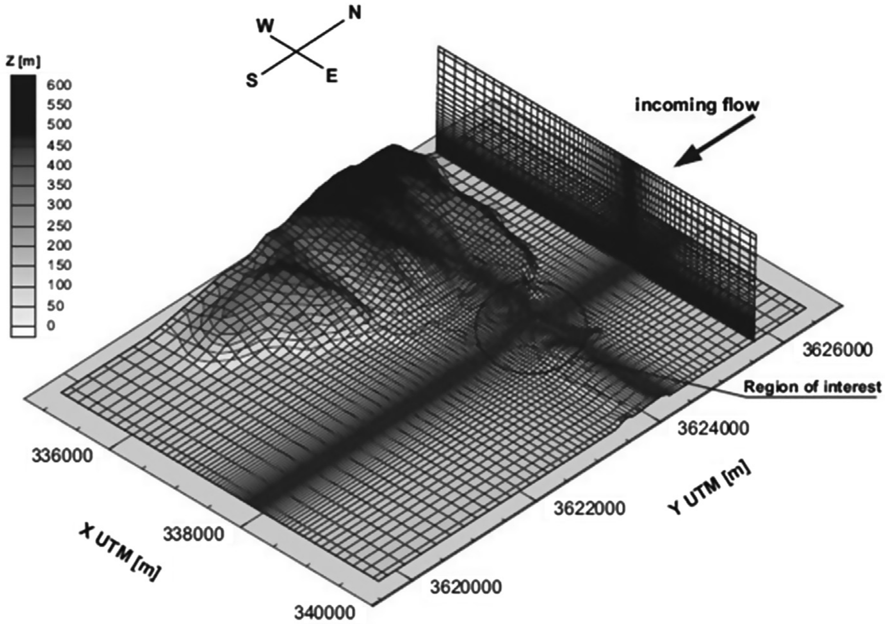

Fig. 1. Complex Terrain Mesh [4].

Table 1
Wind flow analysis comparison.

Ref.	Methods	Program(s) used	Contribution	Blade model	Results
[4]	Coupled CFD and wind simulation	SIMPLE algorithm and WAsP	Commercial CFD and wind sim. coupling	None	Accurate micro-siting
[3]	LES to analyze wake–wake and terrain interference	RIAM-CAMPACT—custom program	Created new model	None	Accurate, within 10% of monthly data
[5]	Urban environment micro-siting with CFD	Ansys CFX with SST	Found best building type	None	Flat roofs best
[6]	Simulated Ekman layer and turbine interaction	Not stated	Effects on ABL from wind farms	Actuator disc	Mixed-height farms better

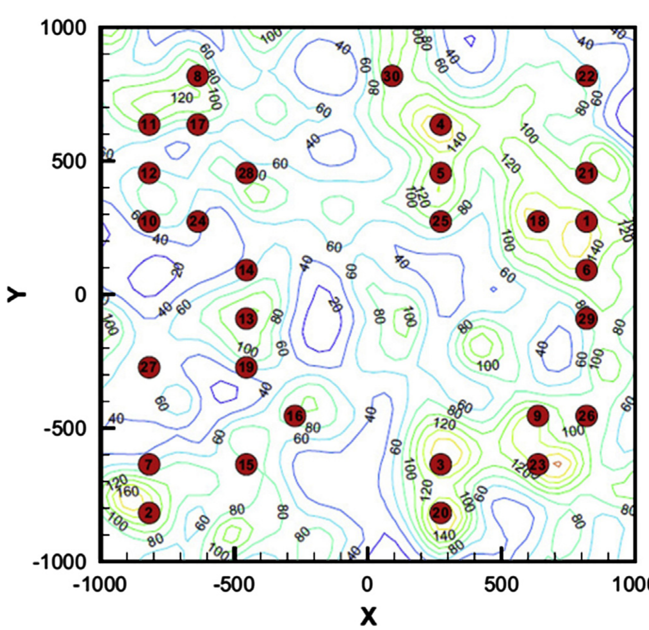

Fig. 2. Optimal layout on complex terrain [9].

interconnections. None of these models consider population data, regulatory restrictions, the effects of transmission, or transmission line locations, but McWilliam et al. [11] explored these factors in layout optimization. Their study provided some insights, but did not consider terrain roughness. Table 2 compares the studies in this subsection by listing their employed methods, fluid, turbine, and cost models, and results.

The focus of this section is using approximation and optimization models for turbine micro-siting. Blade approximation techniques, wind modeling, and algorithmic optimization are new tools in this area for the wind energy industry. Table 3 compares these tools by listing their descriptions, pros, and cons. Layout optimization techniques are valuable, but simulating and predicting wind conditions enable realistic evaluation and analysis of wind farms.

3. ABL and wind modeling

The ability to model, simulate, and predict accurately wind profiles and the ABL is crucial to power output prediction and farm siting and modeling. Numerical methods can fulfill this need. Recent research in this area has focused on improvement of existing wind prediction methods, detailed ABL and flow modeling, and coupling CFD with wind prediction programs and techniques.

3.1. Wind prediction

Modeling and predicting the characteristics of the wind are important to power prediction and siting. Accurate wind prediction models exist [12], but numerical methods can improve the output.

Table 2
Algorithmic and cost-analyzing methods comparison.

Ref.	Methods	Model type	Turbine analysis	Cost model	Results
[7]	Gaussian particle swarm algorithm	Numerical	One type, one height	None	Increases efficiency
[8]	CFD, wake–wake analysis	EllipSys3D, RANS, LES, SIMPLE	Actuator disc, one type, one height	None	Slightly increases power output
[9]	Particle flow field, wake–wake analysis	CFD, program not stated	One type, one height	None	Accurate optimization
[10]	Rotor size and turbine location considered	Particle Swarm Optimization	Analytical wake model, one height, multiple types	Simple, non-comprehensive	Accurate model, verified with exp.
[2]	Coupled CFD and evolutionary algorithm	SIMPLE algorithm CFD and WAsP	One type, one height	Installation cost	Accurate micro-siting
[11]	Wind and population data, regulations, transmission	Gradient optimization with numerical smoothing	None	Regulatory, transmission costs	Good prediction of best farm locations

Table 3
Turbine micro-siting and wind farm layout optimization comparison.

Method	Description	Pros	Cons
Blade Approximation Models	Replaces blade geometry with equivalent model	Simple, fast, accurate far-wake	Inaccurate for near-wake, underestimates power loss
Wind Flow Analysis	Analyzes wind and terrain for micro-siting	Micro-sites well in complex terrain	Slow, turbine geometry often excluded
Algorithmic and Cost-Analyzing Methods	Uses algorithms and cost analysis to micro-site	Creates layouts, considers cost and turbine diameter	Sometimes slow, power predictions not always accurate

An example is a method of predicting wind power potential with a program called Wind Atlas Analysis and Application Program (WAsP) [13]. This program uses wind velocity and direction at a single elevation as input. The method of least squares is used to transform wind data taken at 3 or more elevations into equivalent data taken at one so that these more accurate data could be used as input for WAsP. It is sometimes inaccurate for weak and turbulent winds, but has the advantage of not being dependent on the subjective qualification of the terrain roughness. WAsP works well for developed areas, but developing nations and remote areas require simpler methods. Caixia et al. [14] developed a method of making very short-term wind predictions based on historical wind data in areas where the equipment required for numerical weather prediction is not in place. Wind prediction data is used in short term energy trading and farm power prediction. ARMA, a time series analysis, was combined with a linear model, and predicts wind speed and speed variation better than ARMA alone.

Researchers also have used ABL simulation models to evaluate locations for wind farms. Xu et al. [15] used the MM5 ABL simulation model to assess the potential of a coastal area in Jiangsu, China. The model predicted accurately the distribution of wind velocity and energy in the ABL. A similar study [16] used the WRF model to simulate wind through a wind farm in Xinjiang County, China with complex terrain. This model has different boundary layer schemes suited for different levels of vegetation and terrain types. The quasi-normal scale elimination (QNSE) boundary layer scheme was found to match best the wind data from this arid climate with scarce vegetation. Fig. 3 compares the predicted wind speed produced by the different boundary layer schemes to the observed values. These techniques can help site a wind farm, but micro-siting and evaluation require a more detailed picture of the flow inside the farm.

3.2. CFD ABL and flow modeling

CFD gives a good picture of wind flow through wind farms and can be a tool in evaluating a site. An example is a study [17] which predicted wind velocity profiles in complex terrain using a 3D RANS solver, CRES–flowNS. The model was compared to experimental data and was accurate except in cases of very steep terrain, such as cliffs. This study used traditional CFD methods to model the ABL, but research into more accurate and realistic methods has been done. Usual wall roughness models create inaccurate velocity and turbulence for modeling the ABL, but Parente et al. [18] proposed a modified version of the Richards and Hoxey wall function and k–ε turbulence model to solve this problem. They used FLUENT v6.3 to implement and test the proposed model, and compared results to full scale and wind tunnel tests. The model was capable of creating more accurate ABL simulations, but should not be used to analyze flow around bluff –bodies. This research was continued [19] with the use of FLUENT v13.0 and the open-source CFD program OpenFOAM to simulate the ABL over complex terrain. The researchers found that OpenFOAM better predicted velocity and FLUENT better predicted turbulent kinetic energy when compared to experimental results. They noted that the model was limited by the linearity of the k–ε model.

Another issue with CFD models is the inaccuracy of inflow turbulence. Many turbine simulations use stochastic simulation of inflow turbulence with fixed or deterministic parameters, but studies suggest that these parameters have variation [20]. Saranyasoontorn and Manuel [20] studied turbine load and performance variability when these parameters are treated as random variables. One finding of the study was that variability of turbine load was less than that of the inflow parameters. Fig. 4 shows a turbine overlaid with the turbulence inflow sampling points used in this study. Wind prediction and CFD models are valuable tools by themselves, but when combined, can provide even better insight.

3.3. Wind prediction and CFD coupling

CFD predicts flow around and forces on man-made structures and specific geometry, and wind prediction modeling which predicts large-scale, natural wind phenomena. Coupling the two combines the strengths and scales of each and can be a more powerful tool than each on its own. For example, Rousseau et al. [21] created a Lagrangian stochastic model using MM5 data as input to simulate wind on small scales. Incorporating the MM5 data did not significantly increase computing time.

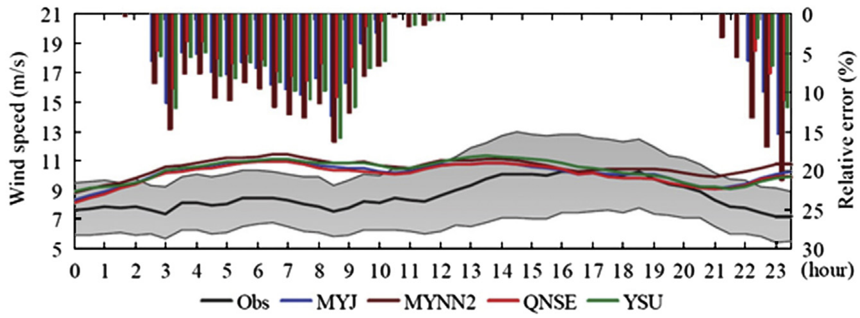

Fig. 3. Mean diurnal wind speed and relative error of BL schemes for February 2008 [16].

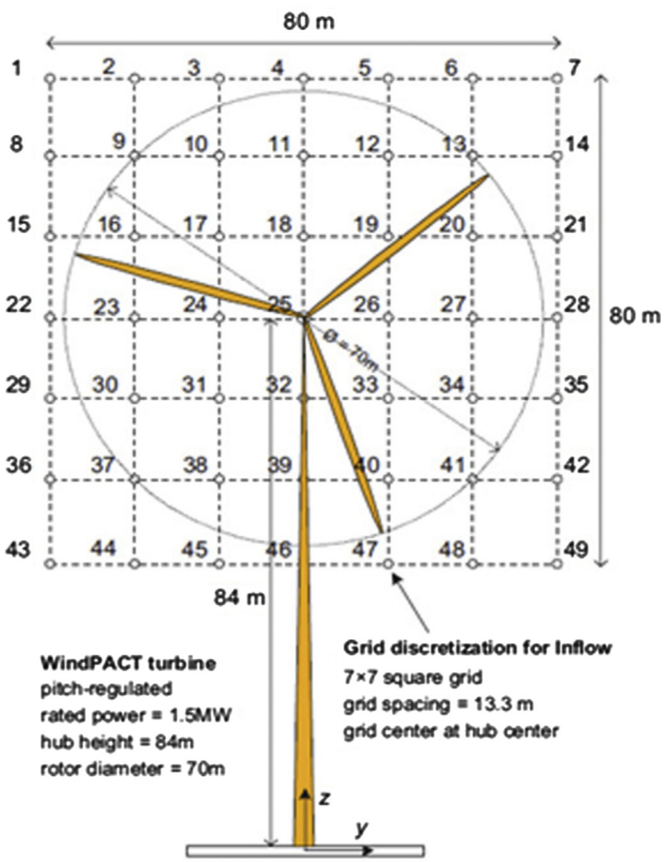

Fig. 4. Turbine overlaid with grid inflow turbulence simulations [20].

The researchers plan to run this model with two different mesh resolutions and validate it with observations in the future. Similarly, Ayotte [22] used the European Wind Atlas and CFD to evaluate wind farms' power potential. This methodology proved to be fairly accurate except in cases where there were abrupt or sharp changes in terrain, such as hills or cliffs.

Researchers also have evaluated the use of commercial CFD programs in coupling. Frank et al. [23] used the CFD program ACUSIM AcuSolve coupled with Numerical Weather Prediction (NWP). The ABL is difficult to resolve, but the proposed technique modeled this region better than traditional methods. This is ongoing research, and efforts to include thermodynamic variables and compare this technique to experimental data are being made. A similar technique [24] couples the National Center for Atmospheric Research (NCAR) community Weather Research and Forecasting (WRF) model with a nested grid/scale scheme and LES to predict wind patterns and profiles. The authors of that study suggested that the model could be improved by the use of sub-grid scale (SGS) techniques and better ground modeling. This model was estimated to be fast enough for real-time predictions. Fig. 5 shows the nested grid technique used in the study.

This section explores recent research of wind and ABL models. Recent research of wind prediction, CFD, ABL, and flow modeling, and wind prediction/CFD coupling are presented as new tools to the industry. Table 4 compares these new tools by listing their employed methods and programs, pros, cons, and results. The subjects considered so far are large-scale in focus, but research in small-scale components of wind farms, such as blades, also has been done.

4. Blade optimization and modeling

CFD and numerical modeling can be used to optimize blade design and control in order to improve the power output, efficiency, and lifespan of wind turbine blades. For example, CFD and structural analysis can be used separately or coupled to predict damage in blades[25]. Power, torque, the interaction between wakes from large numbers of turbines [6], and the effects of blade angle change also can be modeled with numerical techniques [26].

4.1. Blade approximation models

Accurate micro-siting requires that blade geometry and the effects which turbines have on each other be considered. Detailed blade geometry can consume too much computing time, so simplified blade models, such as the actuator disc model, often are employed. The actuator disc model is computationally efficient because it replaces the blade geometry with an infinitely thin disc, which simulates the effects of blades on the flow of the wind [27]. This model is accurate in predicting the wake produced by the turbine at relatively far distances, and it can be used to analyze wake–wake interactions [28,29]. It is one of the only methods that is industrially feasible with current computing technology to simulate a great number to turbines simultaneously [6]. This method normally underestimates power loss from wake–wake interactions, but power output still can be improved through its use [8]. Another approximation model is the actuator line blade model, which replaces the geometry to each blade with an equivalent line [30,31]. The lines rotate and create vorticity and velocity disruptions in the flow field in order to simulate the effects of an actual blade. The actuator line model is illustrated along with large eddy simulation (LES) in Fig. 6. A blade geometry model is valuable, but much more detailed models are required for an analysis with a scale smaller than a wind farm.

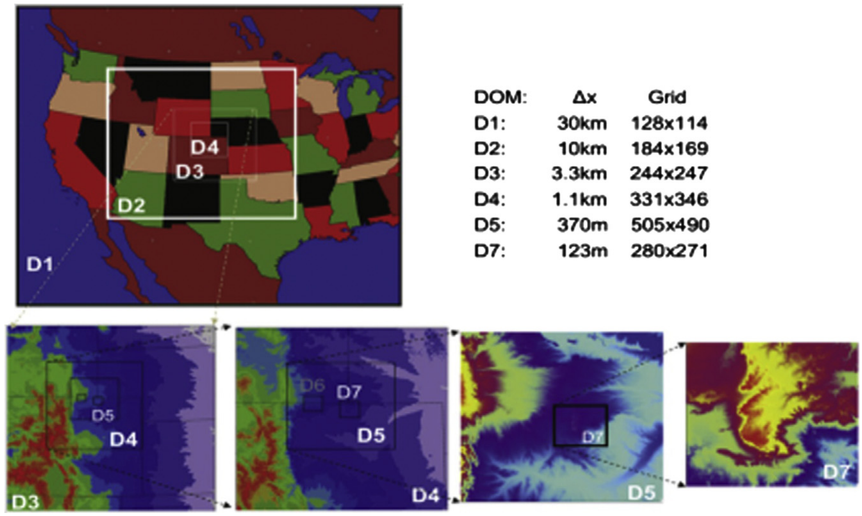

Fig. 5. Nested-grid model domains for simulation of microscale flows [24].

Table 4
ABL and wind modeling comparison.

Ref.	Methods	Program(s) used	Pros	Cons	Results
[13]	Better input for WAsP	WAsP	No subjective terrain roughness	No CFD	Matched experimental data
[14]	ARMA/linear short-term wind prediction	None	Simple and applicable in remote areas	Cannot be used for micro-siting	Accurate wind prediction in areas without numerical wind prediction equipment
[15]	MM5 ABL simulation	MM5	Very accurate wind prediction	No CFD	Matched observations within 7%
[16]	WRF wind simulation	WRF	Different BL schemes available	No CFD	QNSE BL scheme best for arid climates
[17]	Using CFD to predict wind velocity profiles in complex terrain	CRES–flowNS with k–ω turbulence	Accurate profile and power prediction	Inaccurate for steep terrain, no LES	Agreed with experimental data
[18]	Modified Richards and Hoxey/k–ε to model ABL	FLUENT v6.3	Better than traditional wall roughness models	Not suited for bluff bodies	Good prediction of ABL in simple terrain
[19]	Same model as [18] to analyze complex terrain	FLUENT v13.0 and OpenFOAM	Can be implemented accurately in free program	Limited by linearity of k–ε model	Accurate, different results from the programs
[20]	Inflow turbulence variation	Not stated	Better turbulence model		Turbine load variability < inflow
[21]	MM5 and CFD coupling	MM5 and unstated CFD	No increase in computing time	Unconfirmed with data	Successfully coupled wind prediction software with CFD
[23]	Combines large-scale wind prediction and CFD	ACUSIM AcuSolve	Better ABL simulation		Ongoing research
[22]	Combines European Wind Atlas, RANS, and LES	Not stated	Good flow and power prediction	Inaccurate for steep terrain	Fairly accurate for less complex terrain
[24]	Coupled weather prediction/LES, nested scales for micro-siting	NCAR-WRF, RTFDDA	Combines small-and large-scale models, fast	Compared with only one location	Accurate compared to limited data

Fig. 6. Actuator line blade model with large eddy simulation (LES) wake model [30].

4.2. Blade design and analysis

CFD and Finite Element Analysis (FEA) can be used to design complex blades that would not be feasible otherwise. An example is a study [1] that used Ansys FLUENT to design an atypical water wheel turbine with nozzle and diffuser. Another design analysis [32] focused on bend-twist adaptive blades (BTAB) using analytical/FEA coupled aero-structure simulation. The computer-optimized blade design produced more power than a traditional stall regulated turbine. This method is a useful tool, but design parameters besides FEA are needed because using only FEA can be time consuming.

CFD and FEA also can be used to analyze existing design concepts. Maheri et al. [33] developed an aero-structure code, consisting of an aerodynamic simulator, an adaptive mesh generator, and finite element code. They modeled a horizontal axis wind turbine with adaptive blades and studied the effects of

induced twist on blade loading. Another custom piece of software was developed for designing the optimum shape of multi-MW wind turbine blades and analyzing the performance [34]. It was tested and verified by analyzing an existing NREL blade, and the study yielded close correlation and design improvement suggestions. A similar technique [35] uses Navier–Stokes-based CFD and elastic solvers to simulate the aero-elastic behavior of wind turbine blades in classical flutter. This new approach agrees with previous, tested numerical methods, but it requires less computation time. The turbulence model, and to a lesser degree, the coupling scheme affect the outcome greatest.

Blade designs also can be evaluated on the basis of induced loadings and moments. Yuwei et al. [36] used RANS and detached eddy simulation (DES) models implemented in CFDShip-Iowa v4.5 to simulate flow around rotating wind turbine blades. Both models proved accurate for estimating time-averaged forces and moments, but DES more accurately modeled transient conditions. They explored the effects of blade angle change, and considered the possibility of active, dynamic blade change during operation controlled by computations.

Other researchers have focused on fatigue analysis using numerical methods. Shokrieh and Rafiee [37] used FEA to model the critical zone of a blade and accumulated fatigue damage modeling to estimate the fatigue life of wind turbine blades. They used stiffness degradation to measure damage and stochastic loading to simulate wind conditions. Fig. 7 shows the mesh and model they used. Cárdenas et al. [25] also used realistic wind loads to predict damage progression in composite wind turbine blades. Usual methods are cumbersome and require manual intervention, but their method employs a simpler, reduced order model. Their thin-wall beam model uses a one-dimensional model of a blade, greatly reducing computing time. They used a traditional finite element analysis (FEA) to validate the model. Analyzing blade designs on the basis of loads and stresses is useful, but blade angle is another aspect which can be analyzed and optimized.

4.3. Blade angle optimization

Blade pitch greatly affects the power output and efficiency of turbines, and simulations can provide useful insights. As an example, Rajakumar [26] formulated power prediction equations for a horizontal axis wind turbine using blade element momentum theory (BEM), RANS, and the k–ω Shear Stress Transport (SST) turbulence model. The predictions were verified with Ansys FLUENT and the experimental data. The angle of attack was the most significant contributing factor to the power coefficient, and in order to maintain a maximum power coefficient, the angle of attack must vary with the wind velocity. This approach reduced much of the computational effort required by other methods. Two other studies researched the relationship between blade angle and power output using CFD. One used the SIMPLE CFD algorithm with the QUICK interpolating scheme and 1st-order upwind, k–ε turbulence [38], and the other used CFD code based on BEM implemented in FLUENT v6.3.26 [39]. Optimal angles of attack for different wind speeds were calculated in these studies. An example plot, which shows the torque curves and optimum blade angle at different wind speeds, is given in Fig. 8.

The angle of each individual blade can be changed independently to increase efficiency. This type of control is called individual pitch control (IPC). Shen et al. [40] studied the IPC system and fatigue loads on wind turbine blades using a free wake model. IPC greatly reduced the fatigue from asymmetrical, fluctuating effects, such as wind shear. The lifting surface model, used to simulate the blades, does not take into account compressible or viscous effects, so power prediction was not always accurate. The model still proved useful as an aid in IPC.

Using IPC and varying the pitch of all blades simultaneously also is a subject of study. Hwang et al. [41] modeled this control scheme on a cycloidal marine turbine. They verified the numerical predictions with experimental data, and an increase in power output of 25% was achieved by the adoption of IPC. They used the CFD program STAR-CD with moving mesh and the arbitrary sliding interface (ASI) technique. Modeling the blades of a turbine is useful, but considering the entire turbine geometry and using detailed models can provide better design and optimization insights.

5. Detailed turbine and flow modeling

An important decision in research is choosing the most appropriate model. This section presents recent research in model comparisons, improvements, and applications. Using appropriate, refined CFD and numerical models, performing detailed analysis of turbine performance, and considering the entire turbine geometry provide greater insights than approximated or simplified approaches.

5.1. CFD model comparisons and improvements

Many CFD models are available for wind energy applications, but some models may be better for certain circumstances. Comparing research of CFD models in different applications may help researchers pick the best model for a certain situation. Ameur et al.

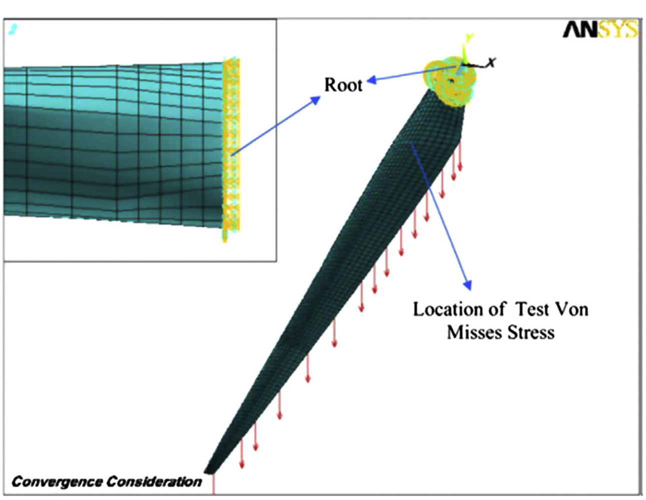

Fig. 7. FEA blade model with loads and boundary conditions [37].

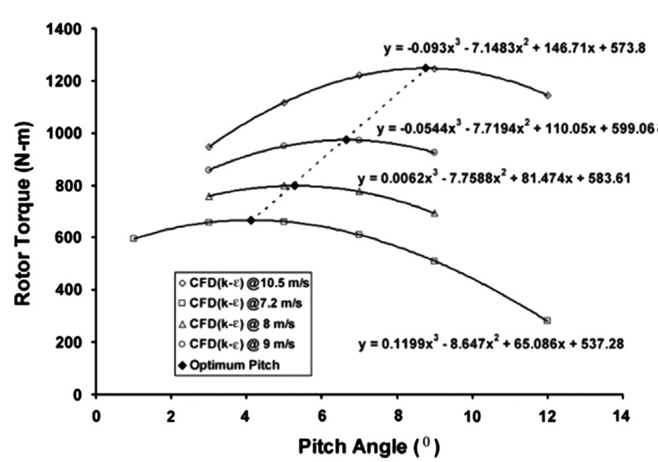

Fig. 8. Computed rotor torque in various wind speeds and pitches [38].

[42] compared different turbulence models in attempting to find the relationship between the wind speed at the nacelle of a wind turbine and the free stream velocity. They used RANS equations with k–ε and k–ω SST turbulence models to analyze the flow around the nacelle while simplifying the rotor model. The atmospheric boundary layer (ABL) was modeled as closely as possible by using a logarithmic velocity profile for the inlet of the model and a special wall model for the ground. The k–ω SST model produced better results compared to experiments without increased computing time. They suggested introducing terrain topography and a more detailed blade model for future studies. More information on optimizing and simplifying the mesh and turbulence models can be found in the 2013 study by Lanzafame et al. [43]. Another comparison study [44] analyzed the performance of a vertical axis wind turbine and the evolution of its wake using CFD. 2D and 3D analyses were used with the k–ω turbulence model and DES. The power output prediction of the 2D/k–ω model was higher than experiments, and that of the 3D/k–ω model was lower. Of the different methods used, the 3D/DES/k–ω model implemented in Ansys FLUENT v12.1 was the most accurate for vortex shedding and turbulent flow prediction. Fig. 9 compares the flow characteristics produced by CFD methods and PIV wind tunnel testing.

Another source of information about CFD models is the study of their failures in certain situations. Moshfeghi et al. [45] conducted this type of analysis by examining the effects of mesh grid spacing near the walls of a fluid model using the SST-k–ω fluid/turbulence model in Ansys CFX v11. Their model was used to study the aerodynamic behavior of a horizontal axis wind turbine. The model did not match the test results well because it did not predict the flow separation correctly.

Other researchers have sought to remedy shortcomings of numerical models. Gao and Hu [46] analyzed the discrepancy between the Reynolds numbers of full-scale and wind-tunnel-scale models of turbines. They used Ansys FLUENT to analyze the performance of a wind-tunnel-scale model. A Reynolds number correction method was developed based on the findings in order to more accurately translate future wind tunnel test data into data for full-scale wind turbines. Studying the differences and shortcomings of different models can be valuable, but successful application of CFD in turbine analysis can yield equally valuable insight.

5.2. Detailed flow and performance prediction

Numerical simulation has become accurate enough to predict flow around and performance of turbines. Such analyses can save time and money by replacing model testing in wind tunnels. An example is a study which modeled a vertical axis wind turbine (VWT) with the CFD software CFdesign 2010 [47]. It used RANS equations in order to discover the turbine's torque characteristics. CFD results aligned with experimental data and showed that the dynamic torque decreased with the tip speed ratio because of the turbine's asymmetrical design. Static torque was found to be consistent with existing VWT designs. Fig. 10 shows the results from the study compared to experimental data.

Similarly, Wang et al. [48] used Ansys FLUENT to predict flow through and the power curve of a small wind turbine with a scoop at low wind speeds. They used the RNG k–ε turbulence model and

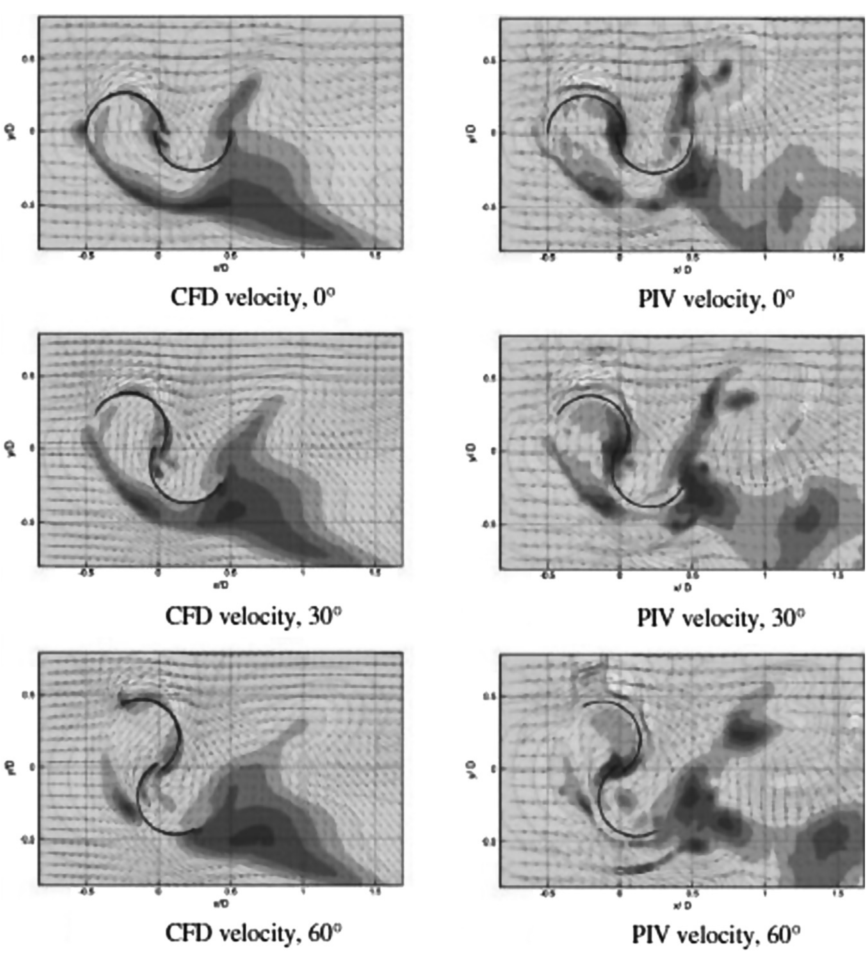

Fig. 9. Comparison of CFD prediction and PIV wind tunnel test [44].

compared the results to wind tunnel tests. The model accurately predicted the flow, and the simulated power curve was within 5% of measured data. Fig. 11 shows a cross sections of the model used in the study.

Szlivka et al. examined flow through another novel design with CFD [49]. They analyzed a dual wind turbine design, consisting of two turbines mounted close together on a single support structure, with Ansys CFX v12.1. The optimum arrangement was found to be turbines offset a half blade diameter with synchronized rotation.

More detailed wake prediction also is a focus of resent research. CHKIR [31] modeled flow through a conventional wind turbine using the actuator line method, and forces were analyzed using the Lagrangian-based free wake method. This model was verified with a wind tunnel test using PIV. The near wake development was predicted correctly, and satisfactory accuracy was obtained for downstream velocity. This method saves computing time compared to other techniques. Grégory Pinon et al. [50] also used a Lagrangian numerical model to predict power and thrust coefficients and simulate the wake created by offshore marine turbines. The simulated wakes were in qualitative and quantitative agreement with experiments, but high resolution simulation using this method requires too much computing power to be economical at this time. The proposed method still is suitable for study of wake–wake interactions. These analyses further the study of detailed flow through turbines, but they neglect the effects of support structures.

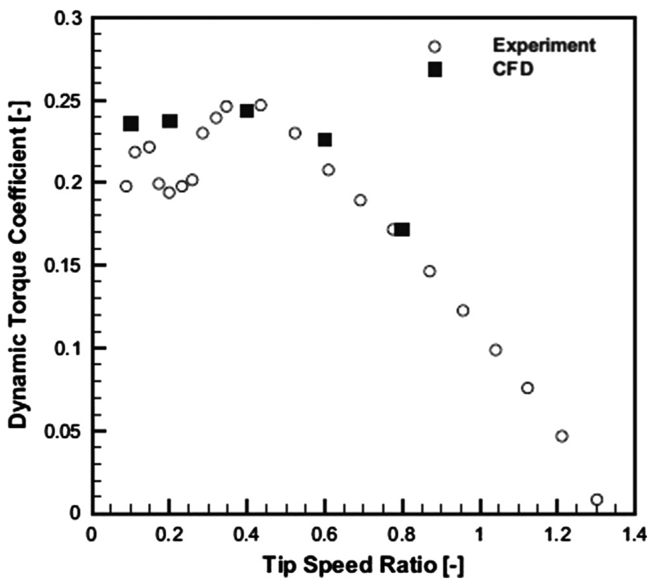

Fig. 10. Validation of numerical predictions with experimental results [47].

5.3. Support structure interference and analysis

The effects of support structures on wind flow can have an impact on turbine performance. Lin and Shieh [51] studied these effects by modeling the flow characteristics, lift forces, and blade-tower interference using CFD on an upwind turbine. The model consisted of separate fixed and moving mesh domains, and it employed an SST k–ω turbulence model. This model successfully predicted flow interference from the tower as the blade passes it. This prediction is not made by other methods which use the potential model. It does not appear that these results were validated by experimental data. Fig. 12 shows cross sections of the blade-tower interference model used in this study.

Tower interference also can cause vibrations. Dolan and Lehn [52] explored the effects of wind shear and tower shadow on torque oscillations for a 3-bladed wind turbine. The model showed the existence of a 3p pulsation, a pulsation at 3 times the rotor frequency, due to wind shear. The study showed how this oscillation is difficult to identify in field tests and experiments. Understanding of oscillation in power output is important to control and power conversion systems. Fig. 13 compares 3p pulsation caused by wind shear to the total 3p pulsation.

Modeling flow around turbines mounted on buildings is another area of interest for recent research. One such research effort used CFD to study the flow characteristics and power output of an increasingly popular type of turbine, the building augmented

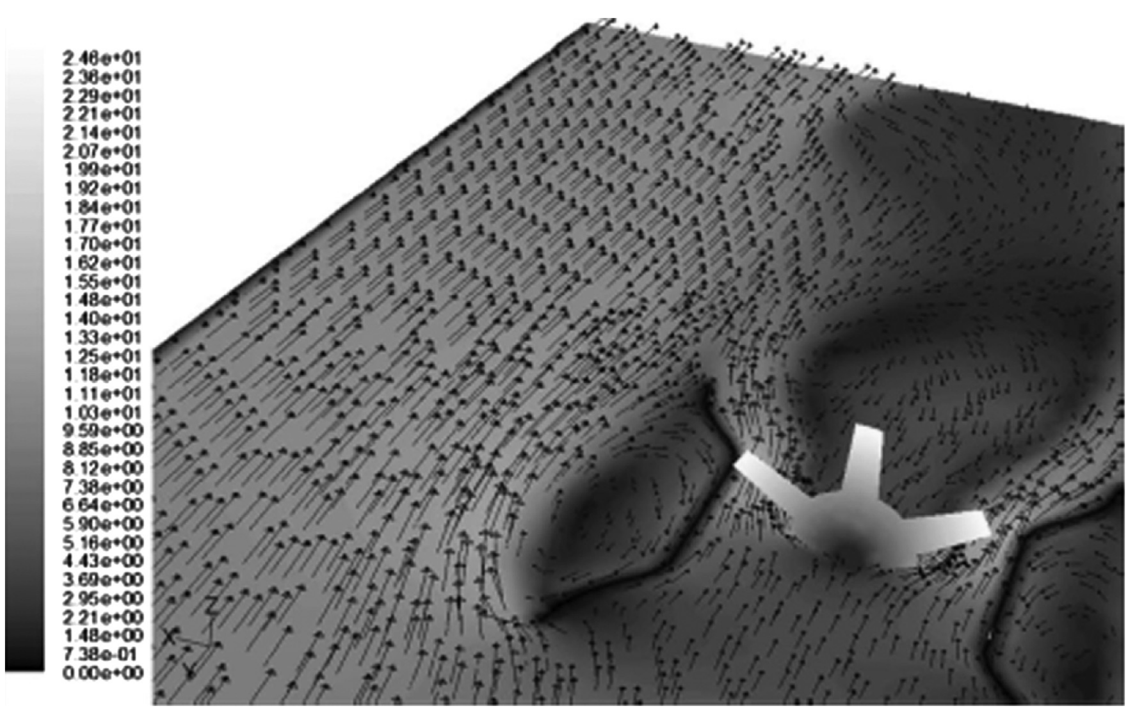

Fig. 11. Contours and vectors of velocity in the scoop [48].

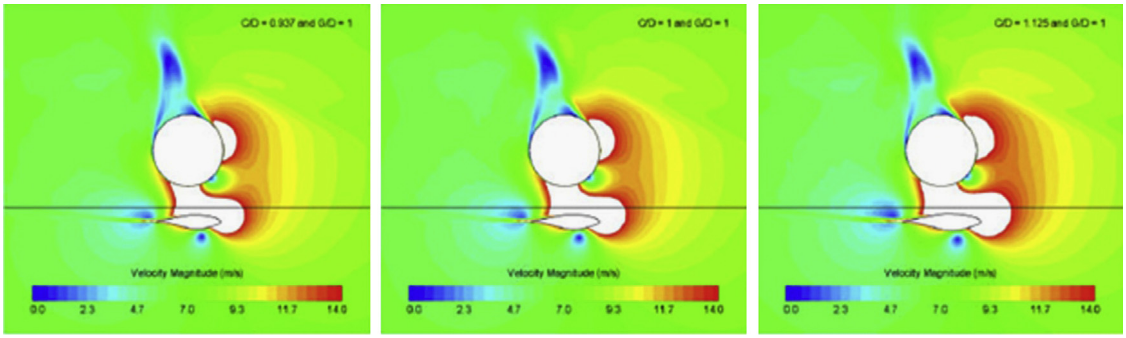

Fig. 12. Velocity fields—the central line is the interface of the tower and the blade domains [51].

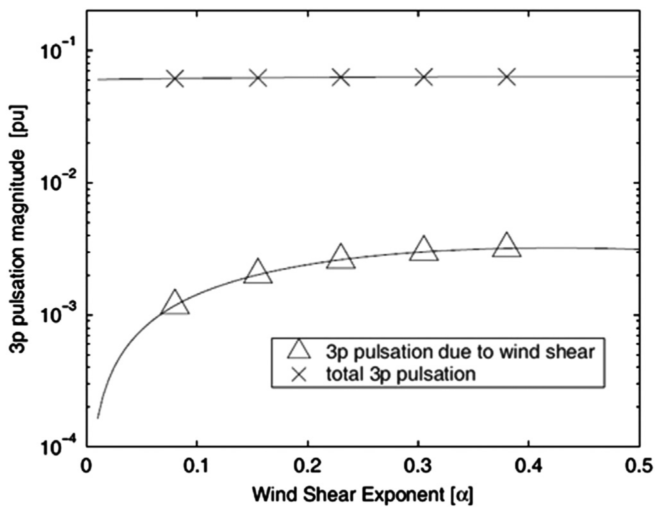

Fig. 13. Relative magnitude (per unit mean torque) of 3p pulsation as a function of wind shear exponent (α) [52].

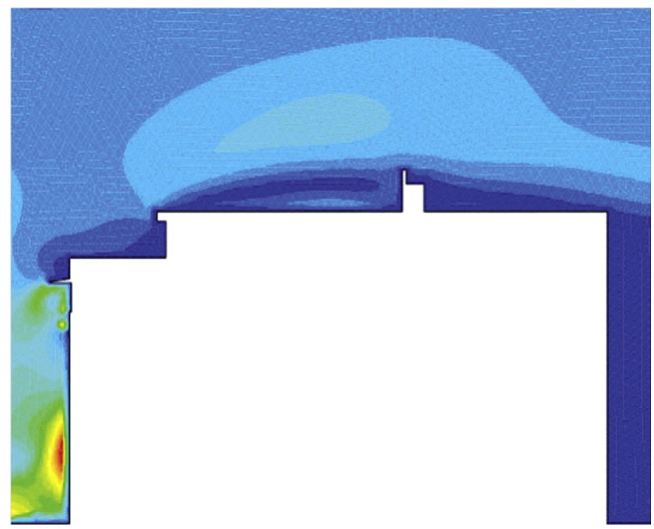

Fig. 14. Contours of velocity magnitude around a structure [54].

wind turbine (BAWT) [53]. Results showed that power output for a BAWT is higher than its standalone counterpart because wind flow is concentrated and accelerated between buildings. The modeled power coefficient exceeded the Betz limit due to this concentrating effect, which is similar to that of ducts or shrouds. This model employed Ansys for CFD analysis. A related study assessed a wind collection and funneling system for application on buildings [54]. Wind tunnel models were constructed using Autocad, Gambit, and FLUENT. This model can be applied to any building and used to scale the appropriate number of turbines. Fig. 14 shows the velocity contour produced by this method around an example structure. The models discussed in this section can analyze the effects of support structures, but not the support structures themselves. They also cannot be used for things such as sound prediction.

6. Support structure and sound simulation

Numerical simulation and analysis of wind turbine support structures and noise generation are two areas of recent interest for researchers. Research of support structures has focused on analysis of new materials, better load simulation, fracture simulation, active structural control systems, and vibration analysis. The two main focuses of numerical noise prediction have been analysis of existing turbines and design of quieter blades.

6.1. Support structure fatigue and force analysis

Numerical simulation can be used to predict the fatigue life and reduce the fatigue loads of wind turbine support structures. An example is a study of the fragility and reliability of steel and concrete support towers for horizontal axis wind turbines [55]. According to the Lagrangian-based fluid model coupled with a solid model, the reinforced, pre-stressed concrete tower performed better than its steel counterpart in long- and short-term strain analysis. This study suggested that concrete towers could be used more predominantly in the future for their superior performance and longer life spans. It also noted that concrete towers can be assembled easily on-site, unlike steel towers, which require difficult and expensive transportation of their segments. Fig. 15 depicts the model and displacement locations in this study.

Extreme load calculation is important to estimating the fatigue life of a large turbine support structures, especially offshore wind turbines exposed to wave impacts. Waves currently are modeled with a linear theory, but more accurate, nonlinear wave models may be more accurate, especially in shallow water where turbines usually are sited. Agarwal and Manuel [56] created a nonlinear wave model and incorporated it into the commonly-used wind turbine simulation software FAST. Predicted long-term loads are significantly higher using this model. Wind loads are higher than wave loads, but wave loads still should be considered. Similarly, Karimirad and Moan [57] performed dynamic response analysis on floating wind turbines that are subjected to wave and wind loads. They used a simplified method for estimating aerodynamic forces to be faster than coupled methods. More comprehensive code validated this simplified approach. This approach is meant to be used for feasibility and pre-engineering studies where effects from rotor dynamics are negligible.

Fracturing caused by cyclic loading is another important design and analysis consideration. Cyclic loading on wind turbine support

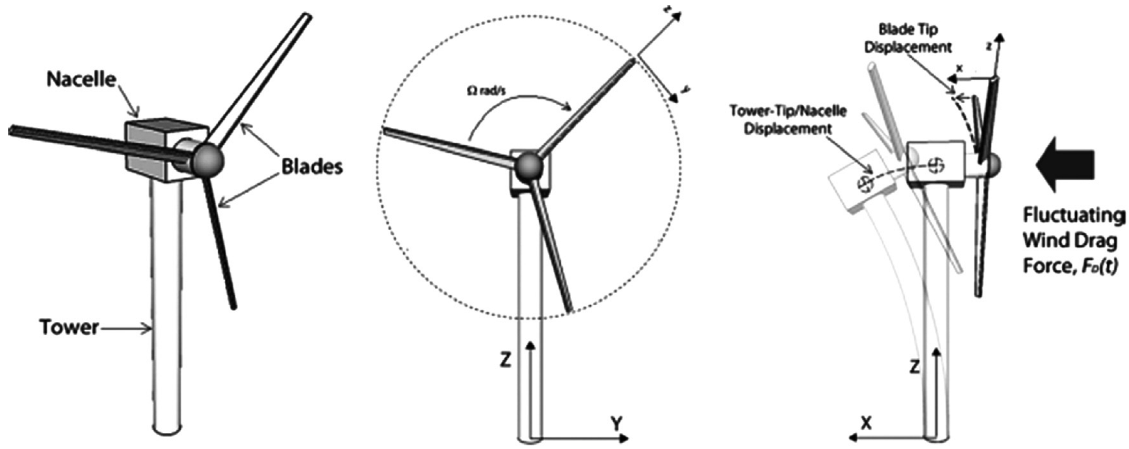

Fig. 15. Sketch of the flap-wise model and coordinate axes [55].

structures causes alternating sates of tension and compression. Existing numerical models can handle fractures caused by tension, but they do not include the strength from the contact effect in compression. Paredes et al. [58] constructed a numerical model which considers the opening and closing of cracks in structural analysis. Their model made use of serial/parallel mixing theory.

Active control systems can minimize support structure forces and stresses. Stewart and Lackner [59] simulated this type of system in large-scale, offshore wind turbines. They compared a limited degree of freedom model, considering actuator dynamics, to a previous work, which used an ideal actuator. They also explored the mechanical aspect of the actuator, and changing the gear ratio reduced the effects of control–structure interaction. FAST-SC was used to implement the model. Minor structural components, such as blades, were neglected.

Considering vibration is another way to minimize stress and loading. Wind turbine towers and their foundations have resonant frequencies, and wind loading causes vibrations in these components. If this induced vibration has the resonant frequency, structural damage may occur. This is an increased concern in offshore turbines due to their increased support structure's length's reducing the resonant frequency [60]. One study used a numerical model to avoid this problem in designing turbine foundations [61]. The researchers used SAP2000 to create a detailed 3D finite element model of the tower, foundation, and pile system. The model successfully predicted the optimal pile size, radius, and material. Fig. 16 shows the simulated relationship between pile radius and the resonant frequency. Another area of recent research is numerical sound prediction.

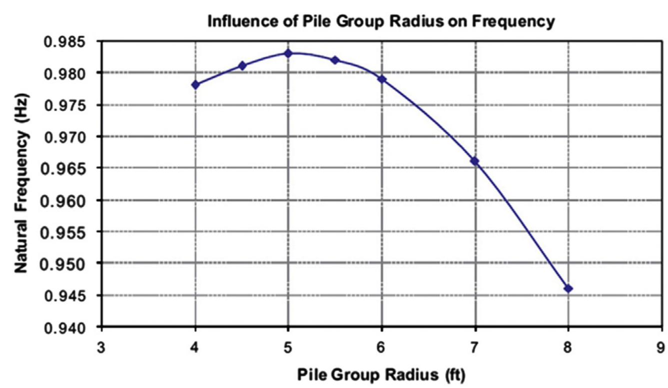

Fig. 16. Frequency variation due to pile radius change [61].

6.2. Numerical noise prediction

The noise generated by wind turbines can be modeled, and reduced, through the use of numerical methods. The first principle-based numerical model for predicting the noise generated by horizontal axis wind (HAWT) turbine blades was developed and validated by Tadamasa and Zangeneh [62]. They used code based on the Ffowcs Williams–Hawkings (FW–H) equation. Ansys CFX v11.0 was used to model the flow around the blades in order to provide the required input for the code. The model components agreed with experimental results. This model can be used as an analysis tool to design quieter blades. A similar analysis study [63] used the fluid–structure simulator, WINFAS, to analyze and predict blade noise and the effect of blade flexibility on the noise level. The researchers found that elastic blades decreased the noise level because they reduced the angle of attack. This study did not consider individual pitch control, stall control, unsteady wind condition, or time dependent wind velocity. Son et al. [64] also used WINFAS along with integrated numerical methods based on Ray theory to predict tonal noise, turbulence ingestion noise, and airfoil self-noise. To determine the noise level on the ground, they considered the effects of air absorption, ground reflection, and diffraction. They also considered the effects of terrain, and found them to be the most important. Fig. 17 shows the noise propagation predicted by this model with and without terrain consideration.

The aforementioned sound prediction studies only analyzed the produced sound, but sound can be used as a design consideration. Göçmen and Özerdem [65] used numerical methods to decrease the noise emission of wind turbine blades while optimizing their aerodynamic performance by changing the shape of the airfoil. The flow analysis tool XFOIL and the airfoil self-noise analysis tool NAFNoise optimized the aerodynamic performance and noise emission separately. This technique reduced noise emission and improved aerodynamic performance in small-scale turbines.

7. Computing trends

All of the methods outlined in this paper have been made possible by increases in computing power and computing techniques. Understanding the basic trends in the advancement of computing is useful for planning and devising research projects in this area.

One trend in numerical simulation research is toward coupling methods and programs with different strengths and scales for greater accuracy, detail, and applicability. FEA and CFD have different

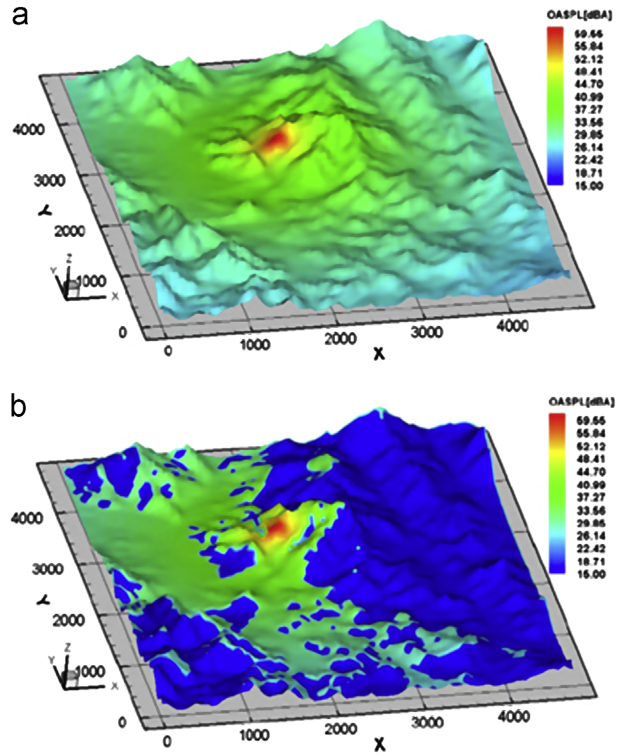

Fig. 17. Wind turbine noise contour with and without terrain consideration [64]. (a) Without terrain effects and (b) With terrain effects.

strengths, and researchers have combined the two to design bend-twist adaptive blades (BTAB) [32]. Examples of coupling models of different scales are two studies which coupled numerical weather prediction (NWP) and CFD to better predict wind profiles and patterns inside wind farms [23,24]. A powerful application of coupling is the combination of CFD, algorithms, cost models, terrain consideration, and regulatory restrictions to optimize the layout of wind farms [2,10,11].

Another trend is toward local and distributed parallel computing techniques. Paredes et al. [58] is an example which uses parallel computing techniques. Advancements in computing technology and solver methods enable linearly-scalable parallel computing networks, which can be used for CFD solutions [66]. Numerical simulation software has taken advantage of new advances in computing technology, and local workstations and servers need to be up-to-date in order to make full use of these tools. Ansys, the maker of FLUENT and CFX, which were used in many of the reviewed studies, has recommendations for servers and workstations used for this type of analysis [67]. Ansys recommends the Intel i7 or AMD Shanghai 8-core processors, 4 to 8 GB of RAM for each core, and the nVIDIA Quadro or ATI FireGL video cards.

8. Conclusion

This paper reviews recent research of numerical simulation as applied to wind energy. Trends in techniques, technology, and application of numerical simulation can be observed from the presented material and related sources. Areas of possible future research also present themselves in the present paper.

Replacing or supplementing experiments with numerical simulation is a feature of resent research. An example is a study [44] that compared CFD and wind tunnel analyses of a turbine's wake. Gao and Hu [46] used CFD to improve the accuracy of wind tunnel test data. Szlivka et al. [49] used CFD to optimize the design of a dual wind turbine instead of resorting to wind tunnel testing. Numerical methods also can give insight into phenomena which are difficult or impossible to observe or measure with experiments. Support structure flow interference is an example of such a phenomenon [51]. Dolan and Lehn [52] researched 3p pulsation due to wind shear that is difficult to observe experimentally. Another example [61] used FEA to predict and minimize vibration damage in turbine support structures.

Areas of possible future research arise from the shortcomings of the presented studies. One such area arises in ABL simulation. Two presented studies [17,22] noted that the employed methods were inaccurate in steep terrain, such as cliffs. Recent research into including cost into wind farm layout optimization [2,10,11] has included wake–wake interactions, terrain, initial cost, turbine size and height, regulatory restrictions, population data, and transmission considerations, but not all simultaneously. Formulating a more complete model may be beneficial.

A small overview of the current trends in computing power and techniques is given. These trends are important to understand when planning a numerical research project. One trend in numerical simulation research is toward coupling methods and programs with different strengths and scales for greater accuracy, detail, and applicability. An example is coupling numerical weather prediction with CFD. Another trend is toward local and distributed parallel computing techniques. Advancements in computing technology and solver methods enable linearly-scalable parallel computing networks, which can be used for CFD solutions. Examples of minimum hardware requirements for workstations and recommendations from companies are given.

References

[1] Wang J-f, Piechna J, MÜLler N. A novel design of composite water turbine using CFD. Journal of Hydrodynamics, Series B. 2012;24:11–6.

[2] Saavedra-Moreno B, Salcedo-Sanz S, Paniagua-Tineo A, Prieto L, Portilla-Figueras A. Seeding evolutionary algorithms with heuristics for optimal wind turbines positioning in wind farms. Renewable Energy 2011;36:2838–44.

[3] Uchida T, Ohya Y. Micro-siting technique for wind turbine generators by using large-eddy simulation. Journal of Wind Engineering and Industrial Aerodynamics 2008;96:2121–38.

[4] Palma JMLM, Castro FA, Ribeiro LF, Rodrigues AH, Pinto AP. Linear and nonlinear models in wind resource assessment and wind turbine micro-siting in complex terrain. Journal of Wind Engineering and Industrial Aerodynamics 2008;96:2308–26.

[5] Ledo L, Kosasih PB, Cooper P. Roof mounting site analysis for micro-wind turbines. Renewable Energy 2011;36:1379–91.

[6] Johnstone R, Coleman GN. The turbulent Ekman boundary layer over an infinite wind-turbine array. Journal of Wind Engineering and Industrial Aerodynamics 2012;100:46–57.

[7] Wan C, Wang J, Yang G, Gu H, Zhang X. Wind farm micro-siting by Gaussian particle swarm optimization with local search strategy. Renewable Energy 2012;48:276–86.

[8] Torres, P., van Wingerden, J.W., Verhaegen, M. Modeling of the flow in wind farms for total power optimization. Control and Automation (ICCA). In: Proceedings of the 9th IEEE International Conference; 2011. p. 963–8.

[9] Song MX, Chen K, He ZY, Zhang X. Wake flow model of wind turbine using particle simulation. Renewable Energy 2012;41:185–90.

[10] Souma Chowdhury JZ, Messac Achille, Luciano Castillo. Unrestricted wind farm layout optimization (UWFLO): investigating key factors influencing the maximum power generation. Renewable Energy 2012;38:16–30.

[11] McWilliam MK, van Kooten GC, Crawford C. A method for optimizing the location of wind farms. Renewable Energy 2012;48:287–99.

[12] Kubik ML, Coker PJ, Barlow JF, Hunt C. A study into the accuracy of using meteorological wind data to estimate turbine generation output. Renewable Energy 2013;51:153–8.

[13] Zeljko Ðurisic JM. A model for vertical wind speed data extrapolation for improving wind resource assessment using WAsP. Renewable Energy 2012;41:407–11.

[14] Caixia, W., Zongxiang, L., Ying, Q. Modeling of wind pattern and its application in wind speed forecasting. Sustainable power generation and supply. In: proceeding of the SUPERGEN '09 International Conference; 2009. p. 1–6.

[15] Xu, X., Chen, Y., Zheng, Y., Bai, X., Wei, M., Liu Y.-a. Observation and Simulation of Wind Energy Resource in Jiangsu Coastal Area. In: proceeding of the Power and Energy Engineering Conference (APPEEC), Asia-Pacific, 2010. p. 1–4.

[16] Chenghai, W., Shuanglong, J., Ju, H., Feimin, Z., Shuanglei, F., Chun, L. Comparing different boundary layer schemes of WRF by simulation the

low-level wind over complex terrain. Artificial Intelligence, Management Science and Electronic Commerce (AIMSEC), In: preceeding of the 2nd International Conference; 2011. p. 6183–8.
[17] Prospathopoulos JM, Politis ES, Chaviaropoulos PK. Application of a 3D RANS solver on the complex hill of Bolund and assessment of the wind flow predictions. Journal of Wind Engineering and Industrial Aerodynamics 2012;107–108:149–59.
[18] Parente CG A, van Beeck J, Benocci C. Improved k–ε model and wall function formulation for the RANS simulation. Journal of Wind Engineering and Industrial Aerodynamics 2011;99:267–78.
[19] Balogh M, Parente A, Benocci C. RANS simulation of ABL flow over complex terrains applying an Enhanced k-ε model and wall function formulation: implementation and comparison for fluent and OpenFOAM. Journal of Wind Engineering and Industrial Aerodynamics 2012;104–106:360–8.
[20] Saranyasoontorn K, Manuel L. On the propagation of uncertainty in inflow turbulence to wind turbine loads. Journal of Wind Engineering and Industrial Aerodynamics 2008;96:503–23.
[21] Rousseau, A., Bernardin, F., Bossy, M., Salameh, T., Drobinski, P. Stochastic particle method applied to local wind simulation. Clean electrical power. In: proceeding of the 2007 ICCEP '07 International Conference; 2007. p. 526–8.
[22] Ayotte KW. Computational modelling for wind energy assessment. Journal of Wind Engineering and Industrial Aerodynamics 2008;96:1571–90.
[23] Frank J, Zajaczkowski SEH, Schmehl Kerrie J. A preliminary study of assimilating numerical weather prediction data into computational fluid dynamics models for wind prediction. Journal of Wind Engineering and Industrial Aerodynamics 2011;99:320–9.
[24] Yubao Liu TW, Liu Yuewei, Vincent Claire, Wu Wanli, Mahoney Bill, Swerdlin Scott, Parks Keith, Jennifer Boehnert. Simultaneous nested modeling from the synoptic scale to the LES scale for wind energy applications. Journal of Wind Engineering and Industrial Aerodynamics 2011;99:308–19.
[25] Cárdenas D, Elizalde H, Marzocca P, Gallegos S, Probst O. A coupled aeroelastic damage progression model for wind turbine blades. Composite Structures 2012;94:3072–81.
[26] Rajakumar S, Ravindran D. Iterative approach for optimising coefficient of power, coefficient of lift and drag of wind turbine rotor. Renewable Energy 2012;38:83–93.
[27] Fernando Porte-Agel Y-TW, Lu Hao, Conzemius Robert J. Large-eddy simulation of atmospheric boundary layer flow through wind turbines and wind farms. Journal of Wind Engineering and Industrial Aerodynamics 2011;99:154–68.
[28] Francesco Castellania AV. An application of the actuator disc model for wind turbine wakes calculations. Applied Energy 2012.
[29] Castellani F, Vignaroli A. An application of the actuator disc model for wind turbine wakes calculations. Applied Energy 2013;101:432–40.
[30] Churchfield MJ, Lee S, Michalakes J, Moriarty PJ. A numerical study of the effects of atmospheric and wake turbulence on wind turbine dynamics. Journal of Turbulence 2012;13:N14.
[31] CHKIR S. Unsteady Loads Evaluation for a Wind Turbine Rotor using free wake method. Energy Procedia 2011;6:777–85.
[32] Maheri A, Noroozi S, Vinney J. Application of combined analytical/FEA coupled aero-structure simulation in design of wind turbine adaptive blades. Renewable Energy 2007;32:2011–8.
[33] Maheri A, Noroozi S, Toomer CA, Vinney J. WTAB, a computer program for predicting the performance of horizontal axis wind turbines with adaptive blades. Renewable Energy 2006;31:1673–85.
[34] Kim B, Kim W, Lee S, Bae S, Lee Y. Developement and verification of a performance based optimal design software for wind turbine blades. Renewable Energy 2013;54:166–72.
[35] Baxevanou CA, Chaviaropoulos PK, Voutsinas SG, Vlachos NS. Evaluation study of a Navier–Stokes CFD aeroelastic model of wind turbine airfoils in classical flutter. Journal of Wind Engineering and Industrial Aerodynamics 2008;96:1425–43.
[36] Yuwei Li K-JP, Xing Tao, Carrica Pablo M. Dynamic overset CFD simulations of wind turbine aerodynamics. Renewable Energy 2012;37:285–98.
[37] Shokrieh MM, Rafiee R. Simulation of fatigue failure in a full composite wind turbine blade. Composite Structures 2006;74:332–42.
[38] Chalothorn Thumthae TC. Optimal angle of attack for untwisted blade wind turbine. Renewable Energy 2009;34:1279–84.
[39] Marco Raciti Castelli AE, Ernesto Benini. The Darrieus wind turbine: Proposal for a new performance prediction model based on CFD. Energy 2011;36:4919–34.
[40] Xin Shen XZ, Zhaohui Du. Wind turbine aerodynamics and loads control in wind shear flow. Energy 2011;36:1424–34.
[41] Hwang IS, Lee YH, Kim SJ. Optimization of cycloidal water turbine and the performance improvement by individual blade control. Applied Energy 2009;86:1532–40.
[42] Khaled Ameur CM, Eecen Peter J. 2D and 3D numerical simulation of the wind-rotor/nacelle interaction in an atmospheric boundary layer. Journal of Wind Engineering and Industrial Aerodynamics 2011;99:833–44.
[43] Lanzafame R, Mauro S, Messina M. Wind turbine CFD modeling using a correlation-based transitional model. Renewable Energy 2013;52:31–9.
[44] Ivan Dobreva FM. CFD and PIV investigation of unsteady flow through Savonius wind turbine. Energy Procedia 2011:711–20.
[45] Mohammad Moshfeghi YJS, Yong Hui Xie. Effects of near-wall grid spacing on SST-K-w model using NREL phase VI horizontal axis wind turbine. Journal of Wind Engineering and Industrial Aerodynamics 2012.
[46] Lin S-Y, Shieh T-H. Study of aerodynamical interference for a wind turbine. International Communications in Heat and Mass Transfer 2010;37:1044–7.
[47] McTavish DF S, Sankar T. Steady and rotating computational fluid dynamics simulations of a novel vertical axis wind turbine for small-scale power generation. Renewable Energy 2012;41:171–9.
[48] Wang LB F, Fletcher J, Whiteford J, Cullen D. The methodology for aerodynamic study on a small domestic wind turbine with scoop. Journal of Wind Engineering and Industrial Aerodynamics 2008;96:1–24.
[49] Szlivka, F., Molnar, I., Kajtar, P., Telekes, G. CFX simulations by twin wind turbine. Electrical and Control Engineering (ICECE). In: proceeding of the International Conference; 2011. p. 5780–3.
[50] Grégory Pinon PM, Germain Grégory, Elie Rivoalen. Numerical simulation of the wake of marine current turbines with a particle method. Renewable Energy 2012;46:111–26.
[51] Shy-Yea Lin T-HS. Study of aerodynamical interference for a wind turbine. International Communications in Heat and Mass Transfer 2010;37:1044–7.
[52] Dolan, D.S.L., Lehn, P.W. Simulation Model of Wind Turbine 3p Torque Oscillations due to Wind Shear and Tower Shadow. In: proceeding of the Power Systems Conference and Exposition 2006. p. 2050–7.
[53] Nak Joon, C., Sang Hyun, N., Jae Sung, K., Sang Min, L., Kyung Chun, K. CFD study on power output and flow characteristics of 110 kW class BAWT. High Performance Computing and Simulation (HPCS). In: proceeding of the International Conference; 2011. p. 859–64.
[54] Saenz-Diez Muro, J.C., Blanco Barrero, J.M., Jimenez Macias, E., Blanco Fernandez, J., Perez de la Parte, M. Optimization of power generation from small wind energy in buildings through modular wind collector for cornice© for capturing and amplification of the vertical wind component. Clean Electrical Power (ICCEP). In: proceeding of the International Conference; 2011. p. 609–13.
[55] Quilligan A, O'Connor A, Pakrashi V. Fragility analysis of steel and concrete wind turbine towers. Engineering Structures 2012;36:270–82.
[56] Agarwal P, Manuel L. Incorporating irregular nonlinear waves in coupled simulation and reliability studies of offshore wind turbines. Applied Ocean Research 2011;33:215–27.
[57] Karimirad M, Moan T. A simplified method for coupled analysis of floating offshore wind turbines. Marine Structures 2012;27:45–63.
[58] Paredes JA, Barbat AH, Oller S. A compression–tension concrete damage model, applied to a wind turbine reinforced concrete tower. Engineering Structures 2011;33:3559–69.
[59] Stewart GM, Lackner MA. The effect of actuator dynamics on active structural control of offshore wind turbines. Engineering Structures 2011;33:1807–16.
[60] Oh K-Y, Kim J-Y, Lee J-S. Preliminary evaluation of monopile foundation dimensions for an offshore wind turbine by analyzing hydrodynamic load in the frequency domain. Renewable Energy 2013;54:211–8.
[61] AlHamaydeh M, Hussain S. Optimized frequency-based foundation design for wind turbine towers utilizing soil-structure interaction. Journal of the Franklin Institute 2011;348:1470–87.
[62] Tadamasa A, Zangeneh M. Numerical prediction of wind turbine noise. Renewable Energy 2011;36:1902–12.
[63] Kim H, Lee S, Son E, Lee S, Lee S. Aerodynamic noise analysis of large horizontal axis wind turbines considering fluid–structure interaction. Renewable Energy 2012;42:46–53.
[64] Son E, Kim H, Kim H, Choi W, Lee S. Integrated numerical method for the prediction of wind turbine noise and the long range propagation. Current Applied Physics 2010;10:S316–9.
[65] Göçmen T, Özerdem B. Airfoil optimization for noise emission problem and aerodynamic performance criterion on small scale wind turbines. Energy 2012;46:62–71.
[66] Hsu M-C, Akkerman I, Bazilevs Y. High-performance computing of wind turbine aerodynamics using isogeometric analysis. Computers and Fluids 2011;49:93–100.
[67] Group S. ANSYS Hardware Support. SimuTech Group; 2012. ⟨http://www.simutechgroup.com/Technical-Support-Services/ansys-hardware-support.html⟩.